Using Sketch Planes and Understanding Sketch Pads

I-DEAS® Tutorials: Fundamental Skills

Learn how to:

- select sketch planes

Before you begin...

Prerequisite tutorials:

1. Getting Started (I-DEAS™ Multimedia Training)

–or–

Introducing the I-DEAS Interface,
Quick Tips to Using I-DEAS
–and–
Creating Parts

2. Sketching and Constraining

3. Dimensioning

4. Building Sections

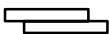To begin this tutorial, make sure you're in the following application and task:

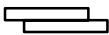[icon] *Design*, *Manufacturing*, or *Simulation*

[icon] *Modeler*

If you saved the model file created in the tutorial "Dimensioning" or "Building Sections," open it now and use the sketch that was created.

[icon] *File*
       Open

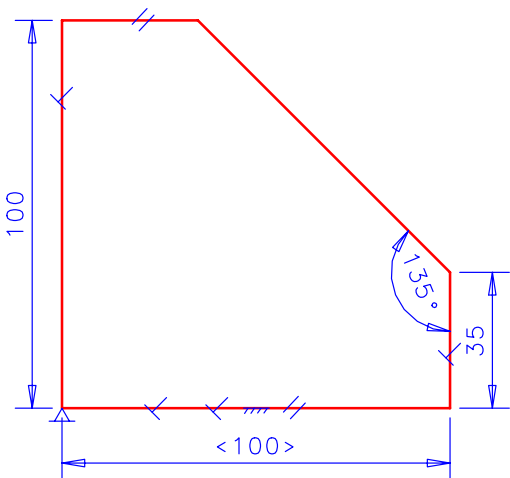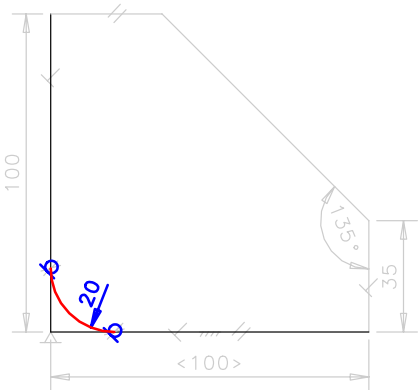Set your units to mm.

[icon] *Options*
       Units

[icon] *mm* (*milli newton*)

If you did not save the model file, or the sketch is no longer on the workbench, create the following sketch before continuing.

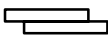Add a fillet in the lower corner. Keep the untrimmed curves.

Hint

Radius: 20

Trim/Extend (toggle off)

Save your model file.

File
    Save

Warning!

If you are prompted by I-DEAS to save your model file, respond:

No

Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

Hint

To reopen your model file to the previous save, press Control-z.

It is important to understand that sketching is always done on a plane. There are two primary methods to orient the sketch plane:

- *Sketch on Workplane*
  - the workplane (this is the default)

- *Sketch in Place*
  - face of the part
  - reference plane associated with the part
 - coordinate systems associated with the part

Sketch on Workplane is primarily used to create a new part. *Sketch in Place* is primarily used to add features to a part.

Another important concept is the sketch pad. You can think of a sketch pad like a pad of paper on which you can sketch. The software uses sketch pads to store and manipulate wireframe geometry.

The software uses three types of sketch pads:

- on the workbench
- on parts
- on features

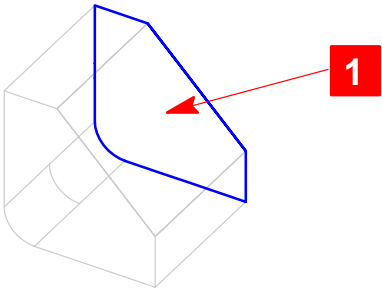First, extrude the geometry on your workbench to create the part shown.
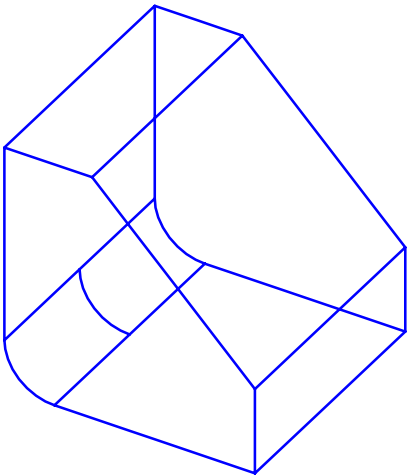
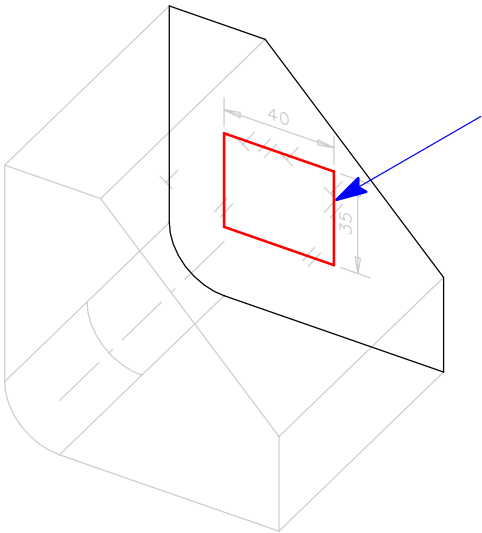**Recovery Point**

File
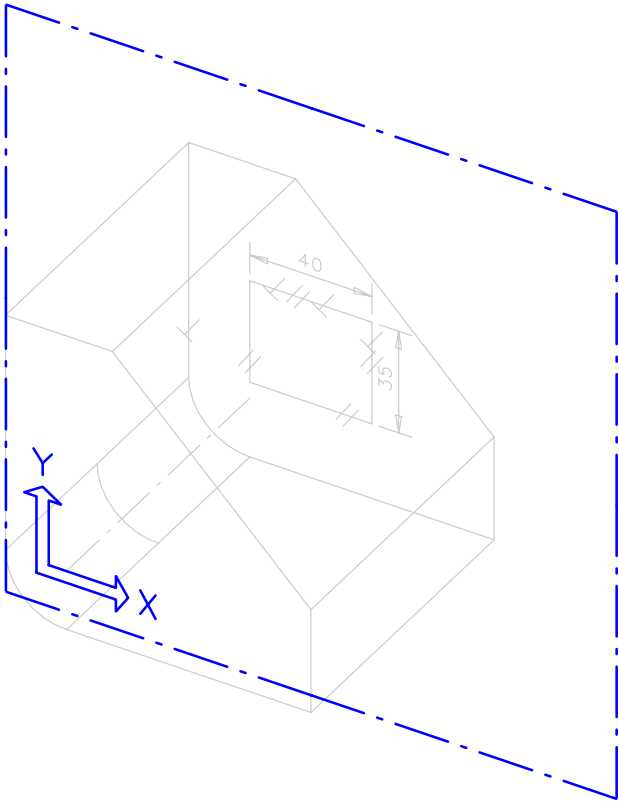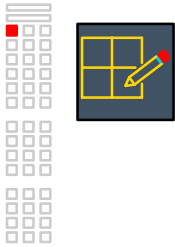   *Save*

Sketch in place on the angled face.
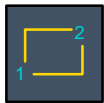

1 pick face

Now, sketch a rectangle on this face. Don't worry about any dimensions that may appear. Wireframe sketched on the part belongs to the part sketch pad.

Revert to sketching on the workplane before continuing.

Sketch a rectangle on the workplane. Don't worry about dimensions.

Before picking a point on the workplane, move the cursor back and forth above the other rectangle.
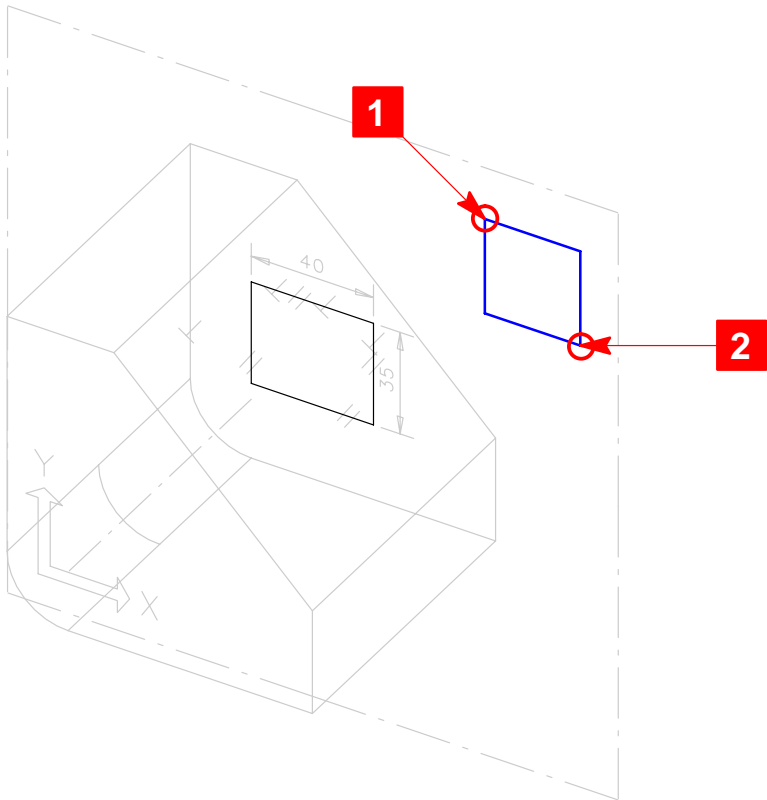
Notice that the Dynamic Navigator does not recognize the geometry on the other rectangle. That's because when you sketch on the workplane, you're also sketching on the workplane sketch pad. Wireframe can't be associated to two different sketch pads.

1

2

 (to deactivate icon)

Although the two rectangles are coplanar, they are not associated with the same sketch pad.
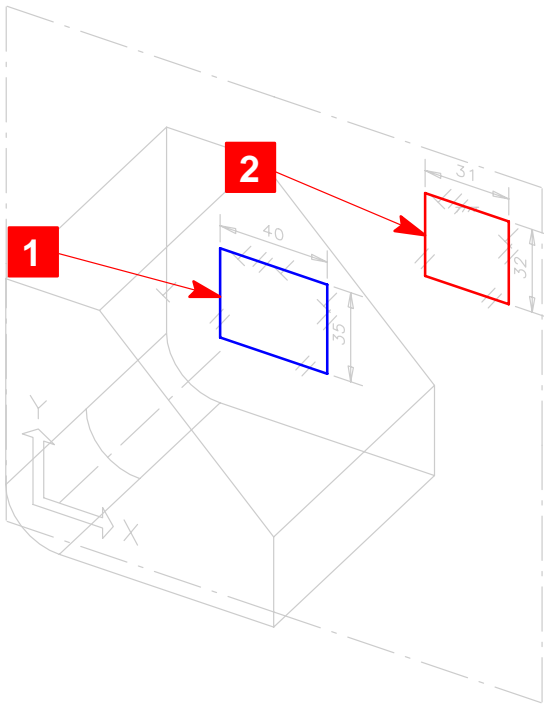
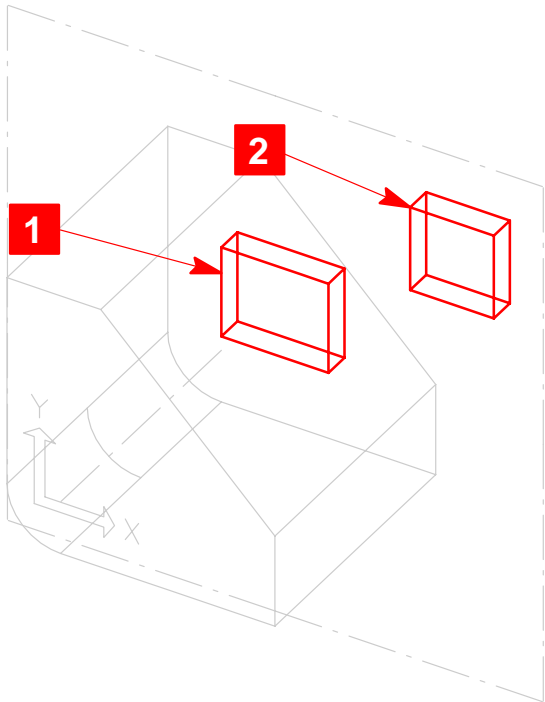Use the *Info* icon and check each rectangle.

1

2 shift-pick

(Done)

Check *I-DEAS List*.

Scroll up in the window to see that one rectangle is associated to the part, and the other with the workbench (notice the part name is "workbench_wireframe").

Extrude each of the rectangles 10 mm. Notice the different default options when you extrude.

Things to notice

In one case, the default is to protrude from the part. In the other case, the default is to create a new part. When you create a feature from wireframe, the software moves the wireframe from the part sketch pad to the feature sketch pad.

The type of sketch pad in which the wireframe is stored determines how you can access and modify the geometry. For more information on sketch pads, see "The Sketch Pad Concept—How Wireframe Geometry Is Stored" in the *Design User's Guide*.

Tutorial wrap-up

You have completed the Using Sketch Planes and Understanding Sketch Pads tutorial.

Delete or put away the parts. These parts are not used in any other tutorials.